Design and manufacturing of plastic injection mould

Content

1 Injection molding

From Wikipedia, the free encyclopedia

Injection molding (British: **moulding**) is a [manufacturing](#) technique for making parts from both [thermoplastic](#) and [thermosetting](#) [plastic](#) materials in [production](#). Molten plastic is injected at high [pressure](#) into a [mold](#), which is the inverse of the product's shape. After a product is designed, usually by an [industrial designer](#) or an [engineer](#), molds are made by a [moldmaker](#) (or toolmaker) from metal, usually either [steel](#) or [aluminium](#), and precision-machined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of [cars](#). Injection molding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. Injection molding typically is capable of tolerances equivalent to an [IT Grade](#) of about 9–14.

The most commonly used thermoplastic materials are polystyrene (low cost, lacking the strength and longevity of other materials), ABS or [acrylonitrile butadiene styrene](#) (a terpolymer or mixture of compounds used for everything from Lego parts to electronics housings), [polyamide](#) (chemically resistant, heat resistant, tough and flexible – used for combs), [polypropylene](#) (tough and flexible – used for containers), [polyethylene](#), and [polyvinyl chloride](#) or PVC (more common in [extrusions](#) as used for pipes, window frames, or as the insulation on wiring where it is rendered flexible by the inclusion of a high proportion of [plasticiser](#)).

Injection molding can also be used to manufacture parts from [aluminium](#) or [brass](#) ([die casting](#)). The melting points of these metals are much higher than those of plastics; this makes for substantially shorter mold lifetimes despite the use of specialized steels. Nonetheless, the costs compare quite favorably to [sand casting](#), particularly for smaller parts.

1.1 History

In 1868 [John Wesley Hyatt](#) became the first to inject hot [celluloid](#) into a mold, producing [billiard balls](#). He and his brother Isaiah patented an injection molding machine that used a plunger in 1872, and the process remained more or less the same until 1946, when James Hendry built the first screw injection molding machine, revolutionizing the plastics industry. Roughly 95% of all molding machines now use screws to efficiently heat, mix, and inject plastic into molds.

1.2 Equipment

Injection molding machines, also known as presses, hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can generate. This pressure keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations.

1.2.1 Mold

Mold (Tool and/or Mold) is the common term used to describe the production tooling used to produce plastic parts in molding.

Traditionally, molds have been expensive to manufacture. They were usually only used in mass production where thousands of parts were being produced. Molds are typically constructed from hardened steel, pre-hardened steel, aluminium, and/or beryllium-copper alloy. The choice of material to build a mold is primarily one of economics. Steel molds generally cost more to construct, but their longer lifespan will offset the higher initial cost over a higher number of parts made before wearing out. Pre-hardened steel molds are less wear resistant and are used for lower volume requirements or larger components. The steel hardness is typically 38-45 on the Rockwell-C scale. Hardened steel molds are heat treated after machining. These are by far the superior in terms of wear resistance and lifespan. Typical hardness ranges between 50 and 60 Rockwell-C (HRC). Aluminium molds can cost substantially less, and when designed and machined with modern computerized equipment, can be economical for molding tens or even hundreds of thousands of parts. Beryllium copper is used in areas of the mold which require fast heat removal or areas that see the most shear heat generated. The molds can be manufactured by either CNC machining or by using Electrical Discharge Machining processes

1.2.2   Design

Molds separate into two sides at a parting line, the *A side*, and the *B side*, to permit the part to be extracted. Plastic resin enters the mold through a *sprue* in the A plate, branches out between the two sides through channels called *runners*, and enters each part *cavity* through one or more specialized *gates*. Inside each cavity, the resin flows around protrusions (called *cores*) and conforms to the cavity geometry to form the desired part. The amount of resin required to fill the sprue, runner and cavities of a mold is a *shot*. When a core *shuts off* against an opposing mold cavity or core, a hole results in the part. Air in the cavities when the mold closes escapes through very slight gaps between the plates and pins, into shallow plenums called *vents*. To permit removal of the part, its features must not overhang one another in the direction that the mold opens, unless parts of the mold are designed to move from between such overhangs when the mold opens. Sides of the part that appear parallel with the direction of *draw* (the direction in which the core and cavity separate from each other) are typically angled slightly with (*draft*) to ease release of the part from the mold, and examination of most plastic household objects will reveal this. Parts with bucket-like features tend to shrink onto the cores that form them while cooling, and cling to those cores when the cavity is pulled away. The mold is usually designed so that the molded part reliably remains on the ejector (B) side of the mold when it opens, and draws the runner and the sprue out of the (A) side along with the parts. The part then falls freely when ejected from the (B) side. *Tunnel gates* tunnel sharply below the parting surface of the B side at the tip of each runner so that the gate is sheared off of the part when both are ejected. Ejector pins are the most popular method for removing the part from the B side core(s), but air ejection, and stripper plates can also be used depending on the application. Most ejector plates are found on the moving half of the tool, but they can be placed on the fixed half if spring loaded. For thermoplastics, coolant, usually water with corrosion inhibitors, circulates through passageways bored through the main plates on both sides of the mold to enable temperature control and rapid part solidification.

To ease maintenance and venting, cavities and cores are divided into pieces, called *inserts*, and subassemblies, also called *inserts*, *blocks*, or *chase blocks*. By substituting interchangeable inserts, one mold may make several variations of the same part.

More complex parts are formed using more complex molds. These may have sections called *slides*, that move into a cavity perpendicular to the draw direction, to form overhanging part features. Slides are then withdrawn to allow the part to be released when the mold opens. Slides are typically guided and retained between rails called *gibs*, and are moved when the mold opens and closes by angled rods called *horn pins* and locked in place by *locking blocks*, both of which move cross the mold from the opposite side.

Some molds allow previously molded parts to be reinserted to allow a new plastic layer to form around the first part. This is often referred to as overmolding. This system can allow for production of one-piece tires and wheels.

2-shot or multi shot molds are designed to "overmold" within a single molding cycle and must be processed on specialized injection molding machines with two or more injection units. This can be achieved by having pairs of identical cores and pairs of different cavities within the mold. After injection of the first material, the component is rotated on the core from the one cavity to another. The second cavity differs from the first in that the detail for the second material is included. The second material is then injected into the additional cavity detail before the completed part is ejected from the mold. Common applications include "soft-grip" toothbrushes and freelander grab handles.

The core and cavity, along with injection and cooling hoses form the mold tool. While large tools are very heavy weighing hundreds and sometimes thousands of pounds, they usually require the use of a forklift or overhead crane, they can be hoisted into molding machines for production and removed when molding is complete or the tool needs repairing.

A mold can produce several copies of the same parts in a single "shot". The number of "impressions" in the mold of that part is often incorrectly referred to as cavitation. A tool with one impression will often be called a single cavity (impression) tool. A mold with 2 or more cavities of the same parts will likely be referred to as multiple cavity tooling. Some extremely high production volume molds (like those for bottle caps) can have over 128 cavities.

In some cases multiple cavity tooling will mold a series of different parts in the same tool. Some toolmakers call these molds family molds as all the parts are not the same but often part of a family of parts (to be used in the same product for example).

1.2.3 Machining

Molds are built through two main methods: standard machining and EDM machining. Standard Machining, in its conventional form, has historically been the method of building injection molds. With technological development, CNC machining became the predominant means of making more complex molds with more accurate mold details in less time than traditional methods.

The electrical discharge machining (EDM) or spark erosion process has become widely used in mold making. As well as allowing the formation of shapes which are difficult to machine, the process allows pre-hardened molds to be shaped so that no heat treatment is required. Changes to a hardened mold by conventional drilling and milling normally require annealing to soften the steel, followed by heat treatment to harden it again. EDM is a simple process in which a shaped electrode, usually made of copper or graphite, is very slowly lowered onto the mold surface (over a period of many hours), which is immersed in paraffin oil. A voltage

applied between tool and mold causes erosion of the mold surface in the inverse shape of the electrode.

1.2.4 Cost

The cost of manufacturing molds depends on a very large set of factors ranging from number of cavities, size of the parts (and therefore the mold), complexity of the pieces, expected tool longevity, surface finishes and many others.

1.3 Injection molding cycle

For the injection molding cycle to begin, four criteria must be met: mold open, ejector pins retracted, shot built, and carriage forward. When these criteria are met, the cycle begins with the mold closing. This is typically done as fast as possible with a slow down near the end of travel. Mold safety is low speed and low pressure mold closing. It usually begins just before the leader pins of the mold and must be set properly to prevent accidental mold damage. When the mold halves touch clamp tonnage is built. Next, molten plastic material is injected into the mold. The material travels into the mold via the sprue bushing, then the runner system delivers the material to the gate. The gate directs the material into the mold cavity to form the desired part. This injection usually occurs under velocity control. When the part is nearly full, injection control is switched from velocity control to pressure control. This is referred to as the pack/hold phase of the cycle. Pressure must be maintained on the material until the gate solidifies to prevent material from flowing back out of the cavity. Cooling time is dependent primarily on the wall thickness of the part. During the cooling portion of the cycle after the gate has solidified, plastication takes place. Plastication is the process of melting material and preparing the next shot. The material begins in the hopper and enters the barrel through the feed throat. The feed throat must be cooled to prevent plastic pellets from fusing together from the barrel heat. The barrel contains a screw that primarily uses shear to melt the pellets and consists of three sections. The first section is the feed section which conveys the pellets forward and allows barrel heat to soften the pellets. The flight depth is uniform and deepest in this section. The next section is the transition section and is responsible for melting the material through shear. The flight depth continuously decreases in this section, compressing the material. The final section is the metering section which features a shallow flight depth, improves the melt quality and color dispersion. At the front of the screw is the non-return valve which allows the screw to act as both an extruder and a plunger. When the screw is moving backwards to build a shot, the non-return assembly allows material to flow in front of the screw creating a melt pool or shot. During injection, the non-return assembly prevents the shot from flowing back into the screw sections. Once the shot has been built and the cooling time has timed out, the mold opens. Mold opening must occur slow-fast-slow. The mold must be opened slowly to release the vacuum that is caused by the injection molding process and prevent the part from staying on the stationary mold half. This is undesirable because the ejection system is on the moving mold half. Then the mold is opened as far as needed, if robots are not being used, the mold only has to open far enough for the part to be removed. A slowdown near the end of travel must be utilized to compensate for the momentum of the mold. Without slowing down the machine cannot maintain accurate positions and may slam to a stop damaging the machine. Once the mold is open, the ejector pins are moved forward, ejecting the part. When the ejector pins retract, all criteria for a molding cycle have been met and the next cycle can begin.

The basic injection cycle is as follows: Mold close – injection carriage forward – inject plastic – metering – carriage retract – mold open – eject part(s) Some machines are run by electric motors instead of hydraulics or a combination of both. The water-cooling channels that assist in cooling the mold and the heated plastic solidifies into the part. Improper cooling can result in distorted molding. The cycle is completed when the mold opens and the part is ejected with the assistance of ejector pins within the mold.

The *resin*, or raw material for injection molding, is most commonly supplied in pellet or granule form. Resin pellets are poured into the feed hopper, a large open bottomed container, which is attached to the back end of a cylindrical, horizontal barrel. A screw within this barrel is rotated by a motor, feeding pellets up the screw's grooves. The depth of the screw flights decreases toward the end of the screw nearest the mold, compressing the heated plastic. As the screw rotates, the pellets are moved forward in the screw and they undergo extreme pressure and friction which generates most of the heat needed to melt the pellets. Electric heater bands attached to the outside of the barrel assist in the heating and temperature control during the melting process.

The channels through which the plastic flows toward the chamber will also solidify, forming an attached frame. This frame is composed of the *sprue*, which is the main channel from the reservoir of molten resin, parallel with the direction of draw, and *runners*, which are perpendicular to the direction of draw, and are used to convey molten resin to the *gate(s)*, or point(s) of injection. The sprue and runner system can be cut or twisted off and recycled, sometimes being granulated next to the mold machine. Some molds are designed so that the part is automatically stripped through action of the mold.

1.4   Molding trial

When filling a new or unfamiliar mold for the first time, where shot size for that mold is unknown, a technician/tool setter usually starts with a small shot weight and fills gradually until the mold is 95 to 99% full. Once this is achieved a small amount of holding pressure will be applied and holding time increased until gate freeze off has occurred, then holding pressure is increased until the parts are free of sinks and part weight has been achieved. Once the parts are good enough and have passed any specific criteria, a setting sheet is produced for people to follow in the future.

Process optimization is done using the following methods. Injection speeds are usually determined by performing viscosity curves. Process windows are performed varying the melt temperatures and holding pressures. Pressure drop studies are done to check if the machine has enough pressure to move the screw at the set rate. Gate seal or gate freeze studies are done to optimize the holding time. A cooling time study is done to optimize the cooling time.

1.5   Molding defects

Injection molding is a complex technology with possible production problems. They can either be caused by defects in the molds or more often by part processing (molding)

| Molding Defects	Alternative name	Descriptions	Causes

Blister	Blistering	Raised or layered zone on surface of the part	Tool or material is too hot, often caused by a lack of cooling around the tool or a faulty heater
Burn marks	Air Burn/ Gas Burn	Black or brown burnt areas on the part located at furthest points from gate	Tool lacks venting, injection speed is too high
Color streaks (US)	Colour streaks (UK)	Localized change of color/colour	Masterbatch isn't mixing properly, or the material has run out and it's starting to come through as natural only
Delamination		Thin mica like layers formed in part wall	Contamination of the material e.g. PP mixed with ABS, very dangerous if the part is being used for a safety critical application as the material has very little strength when delaminated as the materials cannot bond
Flash	Burrs	Excess material in thin layer exceeding normal part geometry	Tool damage, too much injection speed/material injected, clamping force too low. Can also be caused by dirt and contaminants around tooling surfaces.
Embedded contaminates	Embedded particulates	Foreign particle (burnt material or other) embedded in the part	Particles on the tool surface, contaminated material or foreign debris in the barrel, or too much shear heat burning the material prior to injection
Flow marks	Flow lines	Directionally "off tone" wavy lines or patterns	Injection speeds too slow (the plastic has cooled down too much during injection, injection speeds must be set as fast as you can get away with at all times)
Jetting		Deformed part by turbulent flow of material	Poor tool design, gate position or runner. Injection speed set too high.

Polymer degradation		polymer breakdown from hydrolysis, oxidation etc	Excess water in the granules, excessive temperatures in barrel
Sink marks		Localized depression (In thicker zones)	Holding time/pressure too low, cooling time too low, with sprueless hot runners this can also be caused by the gate temperature being set too high
Short shot	Non-fill / Short mold	Partial part	Lack of material, injection speed or pressure too low
Splay marks	Splash mark / Silver streaks	Circular pattern around gate caused by hot gas	Moisture in the material, usually when hygroscopic resins are dried improperly
Stringiness	Stringing	String like remain from previous shot transfer in new shot	Nozzle temperature too high. Gate hasn't frozen off
Voids		Empty space within part (Air pocket)	Lack of holding pressure (holding pressure is used to pack out the part during the holding time). Also mold may be out of registration (when the two halves don't center properly and part walls are not the same thickness).
Weld line	Knit line / Meld line	Discolored line where two flow fronts meet	Mold/material temperatures set too low (the material is cold when they meet, so they don't bond)
Warping	Twisting	Distorted part	Cooling is too short, material is too hot, lack of cooling around the tool, incorrect water temperatures (the parts bow inwards towards the hot side of the tool)

2 Injection molding machine

From Plastics Wiki, free encyclopedia

Injection molding machines consist of two basic parts, an [injection unit] and a [clamping unit]. Injection molding machines differ in both injection unit and clamping unit. The name of the injection molding machine is generally based on the type of injection unit used.
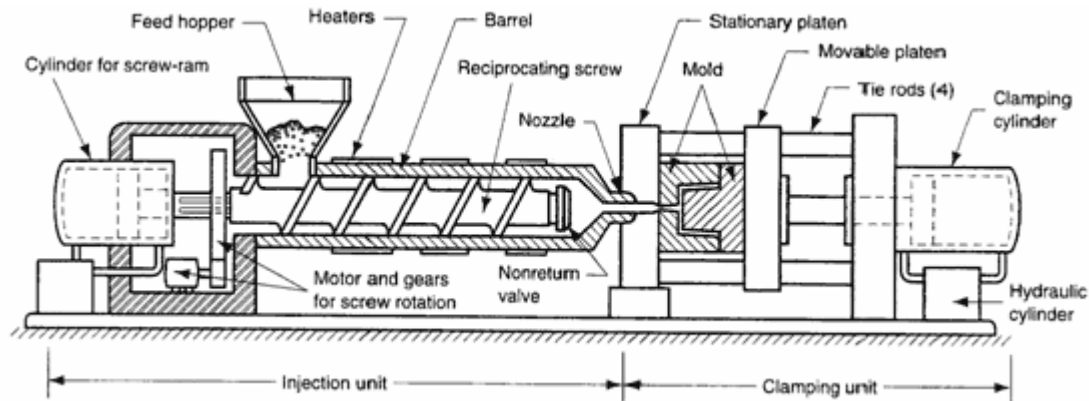
2.1 Types of injection molding machines

Machines are classified primarily by the type of driving systems they use: hydraulic, electric, or hybrid.

2.1.1 Hydraulic

Hydraulic presses have historically been the only option available to molders until [Nissei Plastic Industrial Co., LTD] introduced the first all-electric injection molding machine in 1983. The electric press, also known as [Electric Machine Technology] (EMT), reduces operation costs by cutting energy consumption and also addresses some of the environmental concerns surrounding the hydraulic press.

2.1.2 Electric

Electric presses have been shown to be quieter, faster, and have a higher accuracy, however the machines are more expensive.

2.1.3 Hybrid

Hybrid injection molding machines take advantage of the best features of both hydraulic and electric systems. Hydraulic machines are the predominant type in most of the world, with the exception of Japan.

2.2 Injection unit

The injection unit melts the polymer resin and injects the polymer melt into the mold. The unit may be: [ram fed] or [screw fed].

The **ram fed injection molding machine** uses a hydraulically operated plunger to push the plastic through a heated region. The high viscosity melt is then spread into a thin layer by a "torpedo" to allow for better contact with the heated surfaces. The melt converges at a nozzle and is injected into the mold.

Reciprocating screw A combination melting, softening, and injection unit in an injection molding machine. Another term for the **injection screw**. Reciprocating screws are capable of turning as they move back and forth.

The reciprocating screw is used to compress, melt, and convey the material. The reciprocating screw consists of three zones (illustrated below):

Feed Zone   Transition Metering Zone   Zone

Screw

Barrel
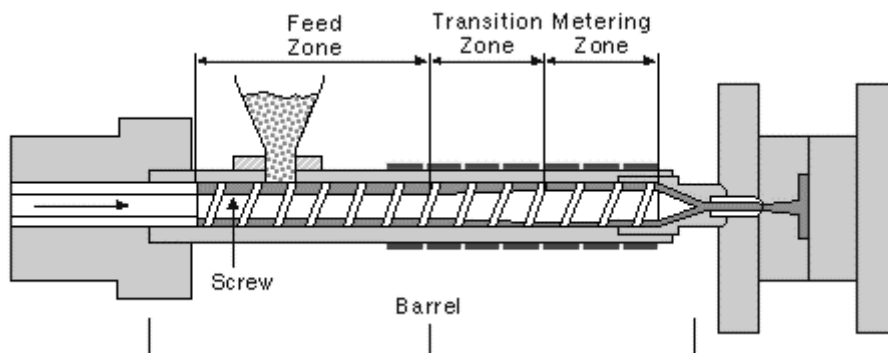
- feeding zone
- compressing zone
- metering zone

While the outside diameter of the screw remains constant, the depth of the flights on the reciprocating screw decreases from the feed zone to the beginning of the metering zone. These flights compress the material against the inside diameter of the barrel, which creates viscous (shear) heat. This shear heat is mainly responsible for melting the material. The heater bands outside the barrel help maintain the material in the molten state. Typically, a molding machine can have three or more heater bands or zones with different temperature settings.

Injection molding reciprocating screw An extruder-type screw rotates within a cylinder, which is typically driven by a hydraulic drive mechanism. Plastic material is moved through the heated cylinder via the screw flights and the material becomes fluid. The injection nozzle is blocked by the previous shot, and this action causes the screw to pump itself backward through the cylinder. (During this step, material is plasticated and accumulated for the next shot.) When the mold clamp has locked, the injection phase takes place. At this time, the screw advances, acting as a ram. Simultaneously, the non-return valve closes off the escape passages in the screw and the screw serves as a solid plunger, moving the plastic ahead into the mold. When the injection stroke and holding cycle is completed, the screw is energized to return and the non-return valve opens, allowing plastic to flow forward from the cylinder again, thus repeating the cycle.

2.2.1   Feed hopper

The container holding a supply molding material to be fed to the screw. The hopper located over the barrel and the feed throat connects them.

2.2.2 Injection ram

The ram or screw that applies pressure on the molten plastic material to force it into the mold cavities.

2.2.3 Injection screw

The [reciprocating-screw](#) machine is the most common. This design uses the same barrel for melting and injection of plastic.

The alternative unit involves the use of separate barrels for plasticizing and injecting the polymer. This type is called a [screw-preplasticizer machine](#) or two-stage machine. Plastic pellets are fed from a hopper into the first stage, which uses a screw to drive the polymer forward and melt it. This barrel feeds a second barrel, which uses a plunger to inject the melt into the mold. Older machines used one plunger-driven barrel to melt and inject the plastic. These machines are referred to as [plunger-type](#) injection molding machines.

2.2.4 Barrel

Barrel is a major part that melts resins transmitted from hopper through screws and structured in a way that can heat up resins to the proper temperature. A band heater, which can control temper atures in five sections, is attached outside the barrel. Melted resins are supplied to the mold passing through barrel head, shot-off nozzle, and one-touch nozzle.

2.2.5 Injection cylinder

Hydraulic motor located inside bearing box, which is connected to injection cylinder load, rotates screw, and the melted resins are measures at the nose of screw. There are many types of injection cylinders that supply necessary power to inject resins according to the characteristics of resins and product types at appropriate speed and pressure. This model employs the double cylinder type. Injection cylinder is composed of cylinder body, piston, and piston load.

2.3 Clamping unit

The clamping unit holds the mold together, opens and closes it automatically, and ejects the finished part. The mechanism may be of several designs, either [mechanical](#), [hydraulic](#) or [hydromechanical](#).

Toggle clamps - a type [clamping unit](#) include various designs. An actuator moves the crosshead forward, extending the toggle links to push the moving platen toward a closed position. At the beginning of the movement, mechanical advantage is low and speed is high; but near the end of the stroke, the reverse is true. Thus, toggle clamps provide both high speed and high force at different points in the cycle when they are desirable. They are actuated either by hydraulic cylinders or ball screws driven by electric motors. Toggle-clamp units seem most suited to relatively low-tonnage machines.

Two clamping designs: (a) one possible toggle clamp design (1) open and (2) closed; and (b) hydraulic clamping (1) open and (2) closed. Tie rods used to guide movuing platens not shown.
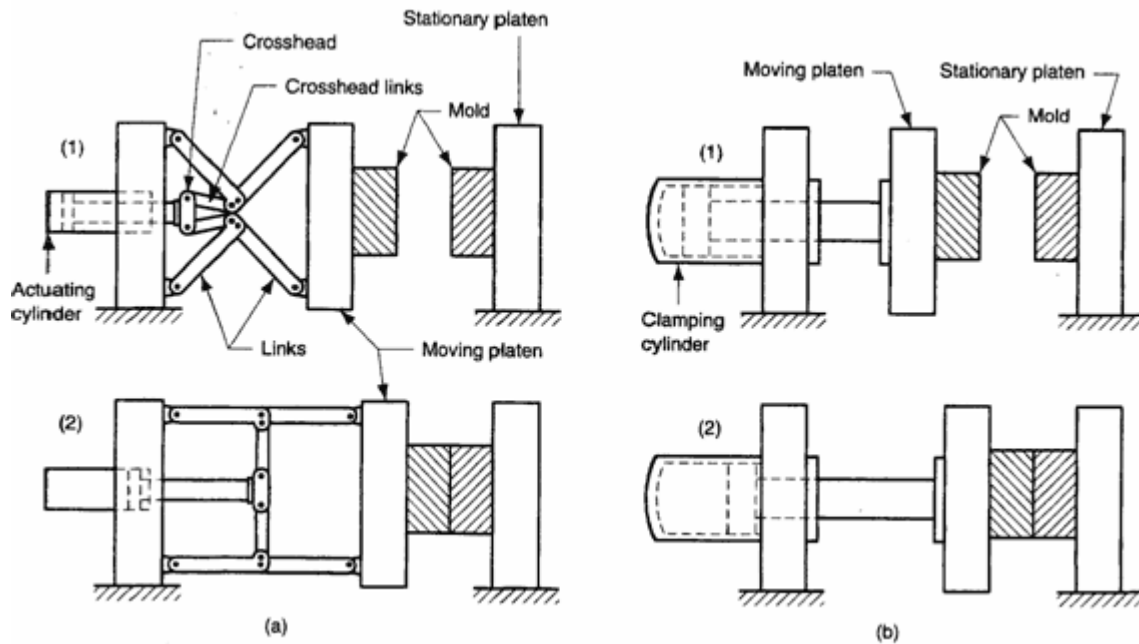
Hydraulic clamps are used on higher-tonnage injection molding machines, typically in the range 1300 to 8900 kN (150 to 1000 tons). These units are also more flexible than toggle clamps in terms of setting the tonnage at given positions during the stroke.

Hydraulic Clamping System is using the direct hydraulic clamp of which the tolerance is still and below 1 %, of course, better than the toggle system. In addition, the Low Pressure Protection Device is higher than the toggle system for 10 times so that the protection for the precision and expensive mold is very good. The clamping force is focus on the central for evenly distribution that can make the adjustment of the mold flatness in automatically.

Hydromechanical clamps - clamping units are designed for large tonnages, usually above 8900 kN (1000 tons); they operate by (1) using hydraulic cylinders to rapidly move the mold toward closing position, (2) locking the position by mechanical means, and (3) using high pressure hydraulic cylinders to finally close the mold and build tonnage.

2.3.1 Injection mold

There are two main types of injection molds: cold runner (two plate and three plate designs) and hot runner – the more common of the runnerless molds.

2.3.2 Injection platens

Steel plates on a molding machine to which the mold is attached. Generally, two platens are used; one being stationary and the other moveable, actuated hydraulically to open and close the mold. It actually provide place to mount the mould. It contains threaded holes on which mould can be mounted using clamps.

2.3.3 Clamping cylinder

A device that actuates the chuck through the aid of pneumatic or hydraulic energy.

2.3.4   Tie Bar

Tie bars support clamping power, and 4 tie bars are located between the fixing platen and the support platen.

3 Injection mould

From Wikipedia, the free encyclopedia

Mold A hollow form or [cavity](#) into which molten plastic is forced to give the shape of the required component. The term generally refers to the whole assembly of parts that make up the section of the molding equipment in which the parts are formed. Also called a tool or [die](#).
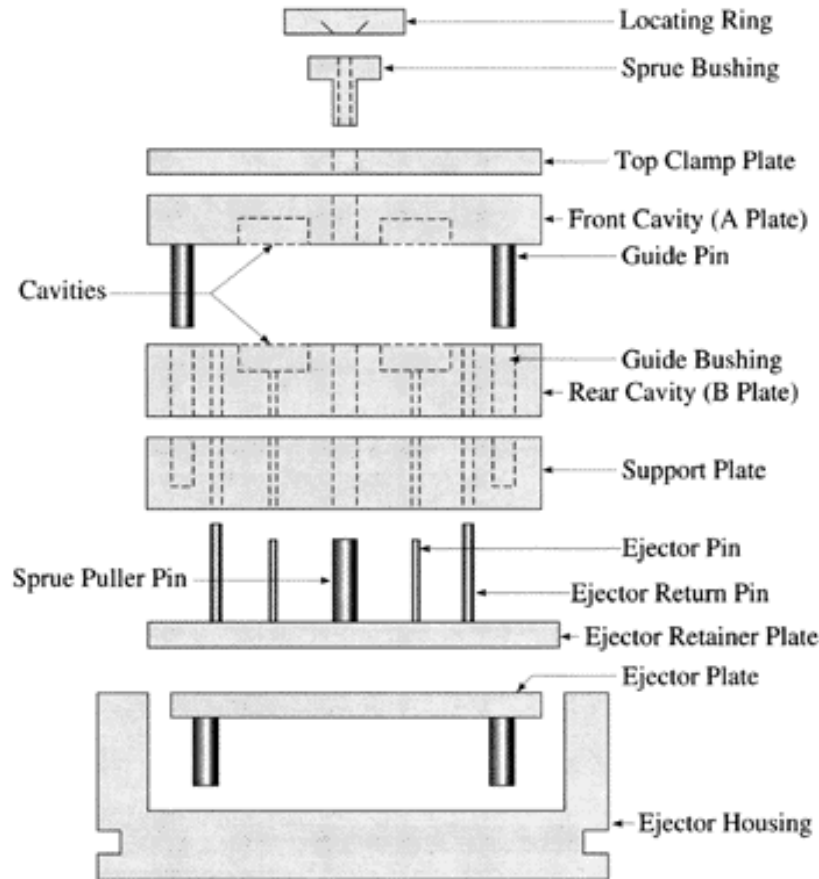
Moulds separate into at least two halves (called the [core](#) and the [cavity](#)) to permit the part to be extracted; in general the shape of a part must be such that it will not be locked into the mould. For example, sides of objects typically cannot be parallel with the direction of draw (the direction in which the core and cavity separate from each other). They are angled slightly; examination of most household objects made from plastic will show this aspect of design, known as draft. Parts that are "bucket-like" tend to shrink onto the core while cooling and, after the cavity is pulled away, are typically ejected using pins. Parts can be easily welded together after moulding to allow for a hollow part (like a water jug or doll's head) that couldn't physically be designed as one mould.

More complex parts are formed using more complex moulds, which may require moveable sections, called slides, which are inserted into the mould to form particular features that cannot be formed using only a core and a cavity, but are then withdrawn to allow the part to be released. Some moulds even allow previously moulded parts to be re-inserted to allow a new plastic layer to form around the first part. This system can allow for production of fully tyred wheels.

Traditionally, moulds have been very expensive to manufacture; therefore, they were usually only used in mass production where thousands of parts are being produced.

Molds require: Engineering and design, special materials, machinery and highly skilled personnel to manufacture, assemble and test them.

Dies and Molds

Labels in diagram (top to bottom): Locating Ring, Sprue Bushing, Top Clamp Plate, Front Cavity (A Plate), Guide Pin, Cavities, Guide Bushing, Rear Cavity (B Plate), Support Plate, Ejector Pin, Sprue Puller Pin, Ejector Return Pin, Ejector Retainer Plate, Ejector Plate, Ejector Housing

Cold-runner mold

Cold-runner mold Developed to provide for injection of thermoset material either directly into the cavity or through a small sub-runner and gate into the cavity. It may be compared to the hot-runner molds with the exception that the manifold section is cooled rather than heated to maintain softened but uncured material. The cavity and core plates are electrically heated to normal molding temperature and insulated from the cooler manifold section.

3.1.1   Types of Cold Runner Molds

There are two major types of cold runner molds: two plate and three plate.

3.1.2   Two plate mold

A two plate cold runner mold is the simplest type of mold. It is called a two plate mold because there is one parting plane, and the mold splits into two halves. The runner system must be located on this parting plane; thus the part can only be gated on its perimeter.

3.1.3   Three plate mold

A three plate mold differs from a two plate in that it has two parting planes, and the mold splits into three sections every time the part is ejected. Since the mold has two parting planes, the runner system can be located on one, and the part on the other. Three plate molds are used

because of their flexibility in gating location. A part can be gated virtually anywhere along its surface.

3.1.4 Advantages

The mold design is very simple, and much cheaper than a hot runner system. The mold requires less maintenance and less skill to set up and operate. Color changes are also very easy, since all of the plastic in the mold is ejected with each cycle.

3.1.5 Disadvantages

The obvious disadvantage of this system is the waste plastic generated. The runners are either disposed of, or reground and reprocessed with the original material. This adds a step in the manufacturing process. Also, regrind will increase variation in the injection molding process, and could decrease the plastic's mechanical properties.

3.1.6 Hot runner mold

Hot-runner mold - injection mold in which the runners are kept hot and insulated from the chilled cavities. Plastic freezeoff occurs at gate of cavity; runners are in a separate plate so they are not, as is the case usually, ejected with the piece.

Hot runner molds are two plate molds with a heated runner system inside one half of the mold.

A hot runner system is divided into two parts: the manifold and the drops. The manifold has channels that convey the plastic on a single plane, parallel to the parting line, to a point above the cavity. The drops, situated perpendicular to the manifold, convey the plastic from the manifold to the part.

3.1.7 Types of Hot Runner Molds

There are many variations of hot runner systems. Generally, hot runner systems are designated by how the plastic is heated. There are internally and externally heated drops and manifolds.

3.1.8 Externally heated hot runners

Externally heated hot runner channels have the lowest pressure drop of any runner system (because there is no heater obstructing flow and all the plastic is molten), and they are better for color changes none of the plastic in the runner system freezes. There are no places for material to hang up and degrade, so externally heated systems are good for thermally sensitive materials.

3.1.9 Internally heated hot runners

Internally heated runner systems require higher molding pressures, and color changes are very difficult. There are many places for material to hang up and degrade, so thermally sensitive materials should not be used. Internally heated drops offer better gate tip control. Internally

heated systems also better separate runner heat from the mold because an insulating frozen layer is formed against the steel wall on the inside of the flow channels.

3.1.10 insulated hot runners

A special type of hot runner system is an insulated runner. An insulated runner is not heated; the runner channels are extremely thick and stay molten during constant cycling. This system is very inexpensive, and offers the flexible gating advantages of other hot runners and the elimination of gates without the added cost of the manifold and drops of a heated hot runner system. Color changes are very easy. Unfortunately, these runner systems offer no control, and only commodity plastics like PP and PE can be used. If the mold stops cycling for some reason, the runner system will freeze and the mold has to be split to remove it. Insulated runners are usually used to make low tolerance parts like cups and frisbees.

3.1.11 Disadvantages

Hot-runner mold is much more expensive than a cold runner, it requires costly maintenance, and requires more skill to operate. Color changes with hot runner molds can be difficult, since it is virtually impossible to remove all of the plastic from an internal runner system.

3.1.12 Advantages

They can completely eliminate runner scrap, so there are no runners to sort from the parts, and no runners to throw away or regrind and remix into the original material. Hot runners are popular in high production parts, especially with a lot of cavities.

Advantages Hot Runner System Over a Cold Runner System include:

- no runners to disconnect from the molded parts
- no runners to remove or regrind, thus no need for process/ robotics to remove them
- having no runners reduces the possibility of contamination
- lower injection pressures
- lower clamping pressure
- consistent heat at processing temperature within the cavity
- cooling time is actually shorter (as there is no need for thicker, longer-cycle runners)
- shot size is reduced by runner weight
- cleaner molding process (no regrinding necessary)
- nozzle freeze and sprue sticking issues eliminated

4 Gate type

As important as selecting the optimal gate size and location is the choice of the type of gate. Gate types can be divided between manually and automatically trimmed gates.

Manually trimmed gates

Manually trimmed gates are those that require an operator to separate parts from runners during a secondary operation. The reasons for using manually trimmed gates are:

- The gate is too bulky to be sheared from the part as the tool is opened.
- Some shear-sensitive materials (e.g., PVC) should not be exposed to the high shear rates inherent to the design of automatically trimmed gates.
- Simultaneous flow distribution across a wide front to achieve specific orientation of fibers of molecules often precludes automatic gate trimming.

Gate types trimmed from the cavity manually include:

- Sprue gate
- Edge gate
- Tab gate
- Overlap gate
- Fan gate
- Film gate
- Diaphragm gate
- External ring
- Spoke or multipoint gate

Automatically trimmed gates

Automatically trimmed gates incorporate features in the tool to break or shear the gate as the molding tool is opened to eject the part. Automatically trimmed gates should be used to:

- Avoid gate removal as a secondary operation.
- Maintain consistent cycle times for all shots.
- Minimize gate scars.

Gate types trimmed from the cavity automatically include:

- Pin gate
- Submarine (tunnel) gates
- Hot runner gates
- Valve gates

4.1   Sprue gate

Recommended for single cavity molds or for parts requiring symmetrical filling. This type of gate is suitable for thick sections because holding pressure is more effective. A short sprue is favored, enabling rapid mold filling and low-pressure losses. A cold slug well should be included opposite the gate. The disadvantage of using this type of gate is the gate mark left on the part surface after the runner (or sprue) is trimmed off. Freeze-off is controlled by the part thickness rather than determined the gate thickness. Typically, the part shrinkage near the sprue gate will be low; shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate.

Dimensions

The starting sprue diameter is controlled by the machine nozzle. The sprue diameter here must be about 0.5 mm larger than the nozzle exit diameter. Standard sprue bushings have a taper of 2.4 degrees, opening toward the part. Therefore, the sprue length will control the diameter of the gate where it meets the part; the diameter should be at least 1.5 mm larger than or approximately twice the thickness of the part at that point. The junction of sprue and part should be radiused to prevent stress cracking

- A smaller taper angle (a minimum of one degree) risks not releasing the sprue from the sprue bushing on ejection.
- A larger taper wastes material and extends cooling time.
- Non-standard sprue tapers will be more expensive, with little gain.
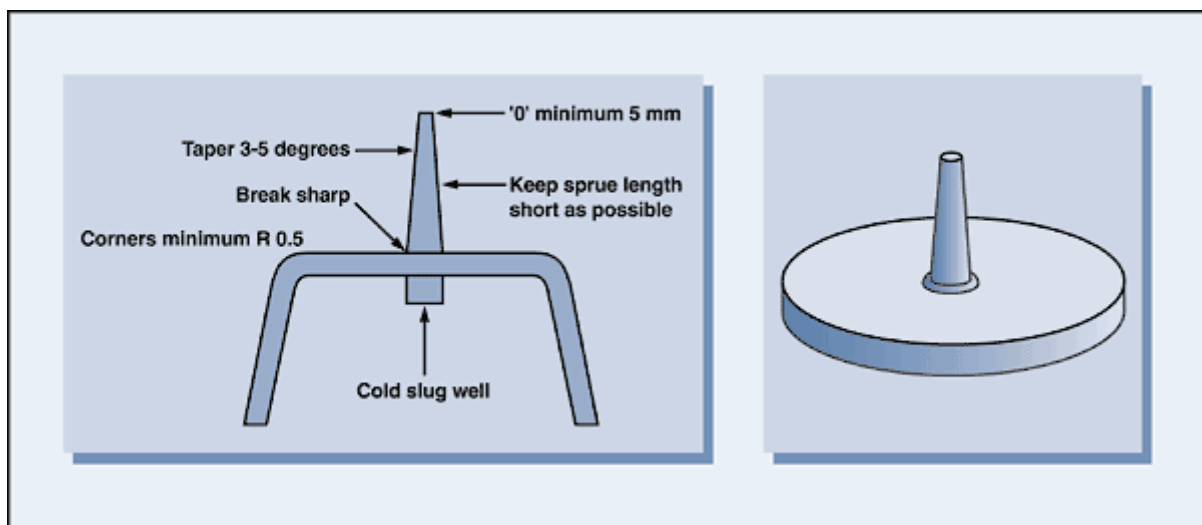
4.2   Edge gate

The edge or side gate is suitable for medium and thick sections and can be used on multicavity two plate tools. The gate is located on the parting line and the part fills from the side, top or bottom.

Dimensions

The typical gate size is 80% to 100% of the part thickness up to 3.5 mm and 1.0 to 12 mm wide. The gate land should be no more than 1.0 mm in length, with 0.5 mm being the optimum.
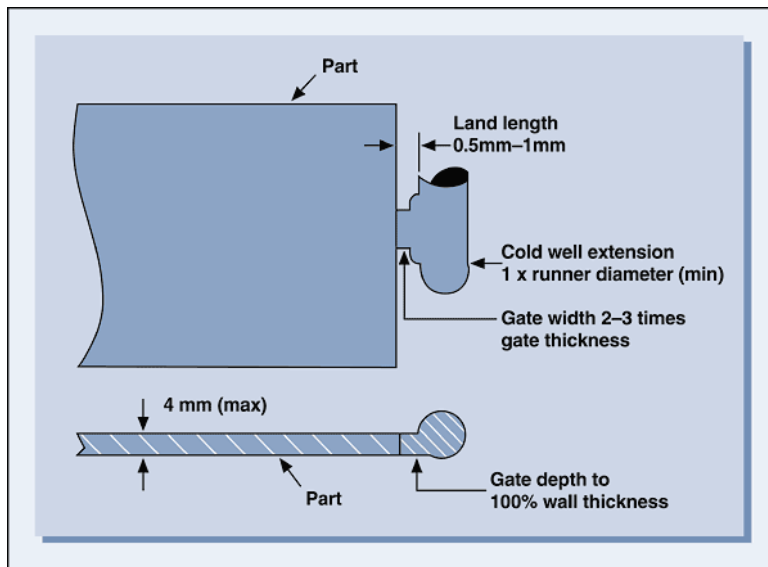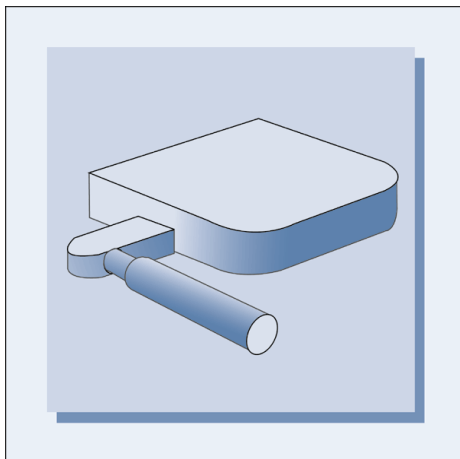

4.3 Tab gate

A tab gate is typically employed for flat and thin parts, to reduce the shear stress in the cavity. The high shear stress generated around the gate is confined to the auxiliary tab, which is trimmed off after molding. A tab gate is often used for molding P.

Dimensions

The minimum tab width is 6 mm. The minimum tab thickness is 75% of the depth of the cavity.

4.4 Overlap gate

An overlap gate is similar to an edge gate, except the gate overlaps the wall or surfaces. This type of gate is typically used to eliminate jetting.

Dimensions

The typical gate size is 10% to 80% of the part thickness and 1.0 to 12 mm wide. The gate land should be no more than 1.0 mm in length, with 0.5 mm being the optimum.
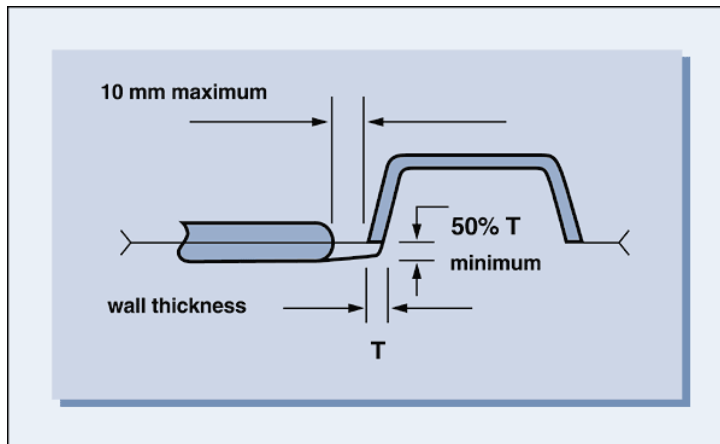
4.5   Fan gate

A fan gate is a wide edge gate with variable thickness. This type is often used for thick-sectioned moldings and enables slow injection without freeze-off, which is favored for low stress moldings or where warpage and dimensional stability are main concerns. The gate should taper in both width and thickness, to maintain a constant cross sectional area. This will ensure that:

- The melt velocity will be constant.
- The entire width is being used for the flow.
- The pressure is the same across the entire width.

Dimensions

As with other manually trimmed gates, the maximum thickness should be no more than 80% of the part thickness. The gate width varies typically from 6 mm up to 25% of the cavity length.

4.6   Film or flash gate

A film or flash gate consists of a straight runner and a gate land across either the entire length or a portion of the cavity. It is used for long flat thin walled parts and provides even filling. Shrinkage will be more uniform which is important especially for fiber reinforced thermoplastics and where warpage must be kept to a minimum.

Dimensions

The gate size is small, typically 0.25mm to 0.5mm thick. The land area (gate length) must also be kept small, approximately 0.5 to 1.0 mm long.
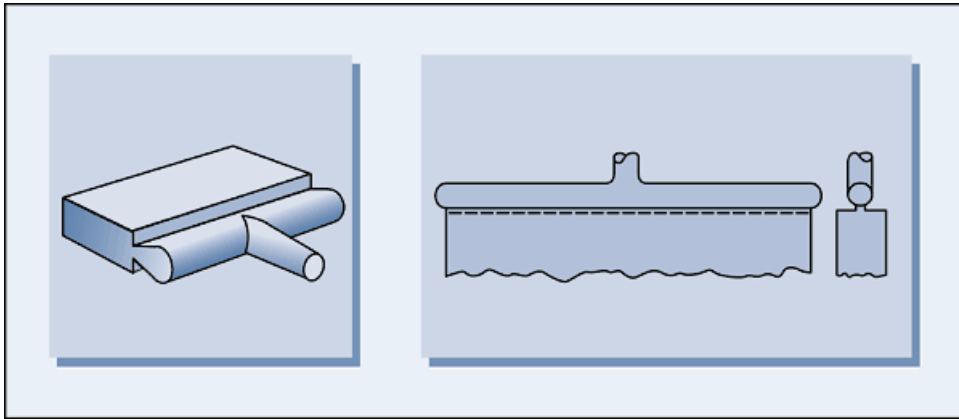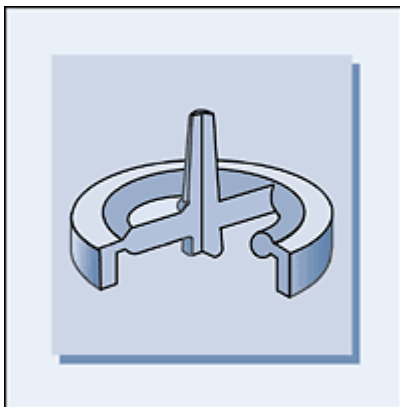
4.7   Diaphragm gate

A diaphragm gate is often used for gating cylindrical or round parts that have an open inside diameter. It is used for single cavity molds that have a small to medium internal diameter. It is used when concentricity is important and the presence of a weld line is not acceptable.

Dimensions

Typical gate thickness is 0.25 to 1.5 mm.
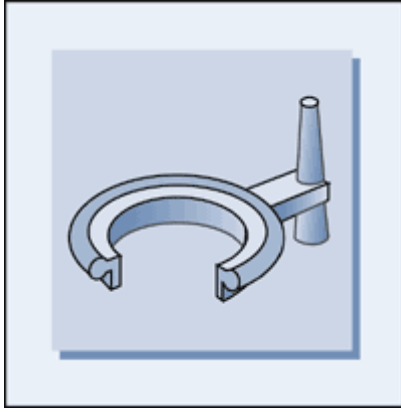
4.8   Internal ring gate.

4.9   External ring gate

This gate is used for cylindrical or round parts in a multicavity mould or when a diaphragm gate is not practical. Material enters the external ring from one side forming a weld line on the opposite side of the runner this weld line is not typically transferred to the part.

Dimensions

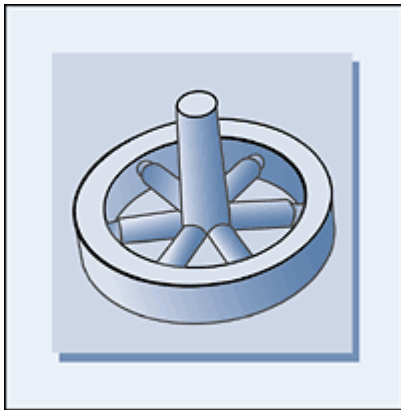Typical gate thickness is 0.25 to 1.5 mm.

4.10 Spoke gate or multipoint gate

This kind of gate is used for cylindrical parts and offers easy de-gating and material savings. Disadvantages are the possibility of weld lines and the fact that perfect roundness is unlikely.

Dimensions

Typical gate size ranges from 0.8 to 5 mm diameter.

4.11 Pin gates

Pin gates are only feasible with a 3-plate tool because it must be ejected separately from the part in the opposite direction The gate must be weak enough to break off without damaging the part. This type of gate is most suitable for use with thin sections. The design is particularly useful when multiple gates per part are needed to assure symmetric filling or where long flow paths must be reduced to assure packing to all areas of the part.

Dimensions

Gate diameters for unreinforced thermoplastics range from 0.8 up to 6 mm. Smaller gates may induce high shear and thus thermal degradation. Reinforced thermoplastics require slightly larger gates > 1 mm The maximal land length should be 1 mm. Advised gate dimensions can be found in the table below.
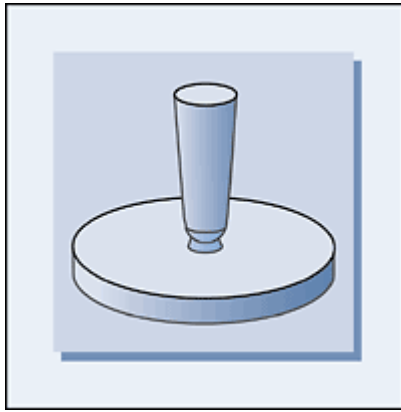
Dimensions of gates (* wall thickness larger than 5 mm should be avoided).

Wall Thickness mm (in)	Gate Diameter / Length mm (in)
0.7 - 1.2 mm (0.02 - 0.05)	0.7 - 1.0 / 0.8 - 1 (0.02 - 0.04 / 0.03 - 0.04)
1.2 - 3.0 mm (0.05 - 0.12)	0.8 - 2.0 / 0.8 - 1 (0.03 - 0.08 / 0.03 - 0.04)
3.0 - 5.0 mm (0.12 - 0.20)	1.5 - 3.5 / 0.9 - 1 (0.06 - 0.14 / 0.04 - 0.04)
$\geq 5.0^*$ (0.20)	3.5 - 6.0 / 0.9 - 1 (0.14 - 0.24 / 0.03 - 0.04)

4.12 Submarine (tunnel) gates

A submarine gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part. The tunnel can be located either in the moving mould half or in the fixed half. A sub-gate is often located into the side of an ejector pin on the non-visible side of the part when appearance is important. To degate, the tunnel requires a good taper and must be free to bend.
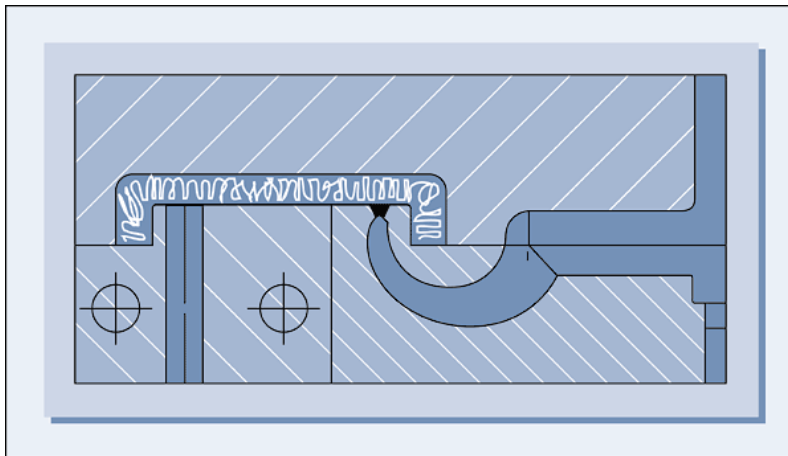
Dimensions

Typical gate sizes 0.8 mm to 1.5 mm, for glass reinforced materials sizes could be larger.
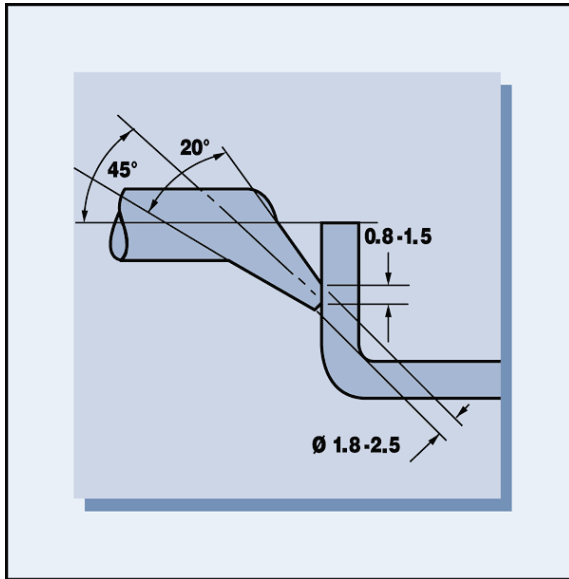
A variation of the tunnel gate design is the curved tunnel gate where the tunnel is machined in the movable mold half. This is not suitable for reinforced materials.

**4.13 Curved tunnel gate.**

4.14 Hot runner gates

Hot runner gates are also known as sprueless gating. The nozzle of a runnerless mold is extended forward to the part and the material is injected through a pinpoint gate. The face of the nozzle is part of the cavity surface; this can cause appearance problems (matt appearance and rippled surface). The nozzle diameter should therefore be kept as small as possible. Most suitable for thin walled parts with short cycle times, this avoid freezing of the nozzle.
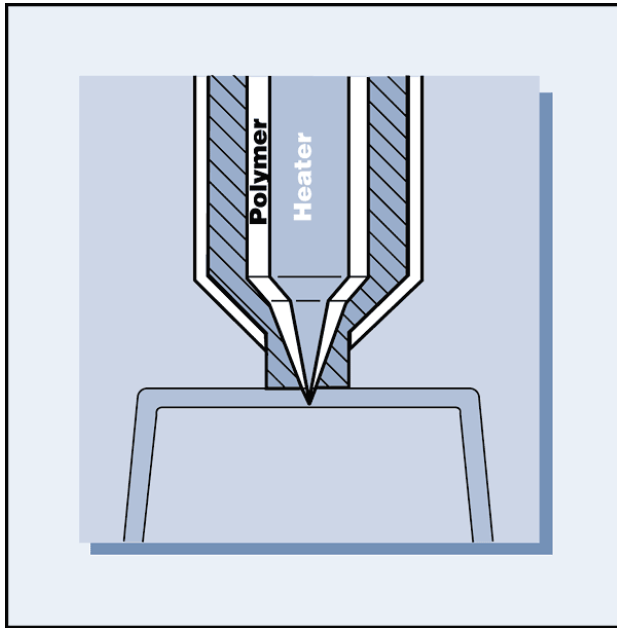
4.15 Valve gates

The valve gate adds a valve rod to the hot runner gate. The valve can be activated to close the gate just before the material near the gate freezes. This allows a larger gate diameter and smoothes over the gate scar. Since the valve rod controls the packing cycle, better control of the packing cycle is maintained with more consistent quality.
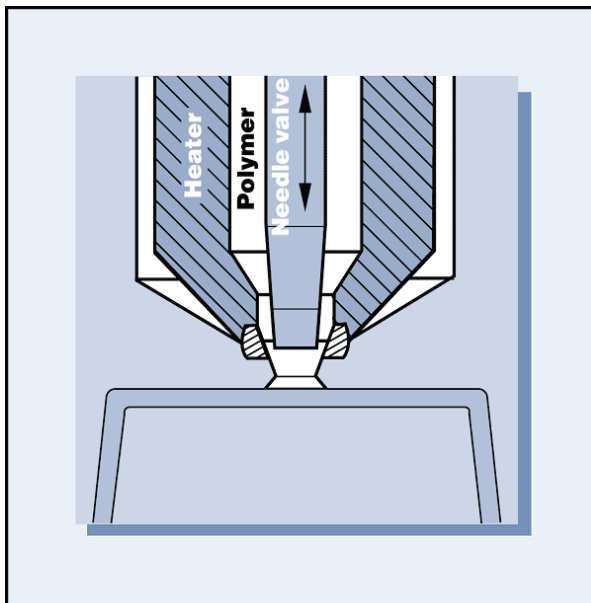
5 Why Molds are Expensive

5.1.1.1 **Introduction:**

The simple answer is that Injection Molds are expensive because they are very complex mechanical systems. Molds require: Engineering and design, special materials, machinery and highly skilled personnel to manufacture, assemble and test them.

The injection molding process is one where molten plastic material is forced into a mold cavity under high pressure. The mold cavity is an exact hollow negative of the part to be produced. In order for the part to be released, the mold must open at the widest place on the part. The molten plastic pressure during injection ranges from 5,000 to over 20,000 psi. This pressure multiplied by the area of the part gives rise to huge forces seeking to open the mold. The mold must be constructed to withstand the very high clamping forces exerted by the injection molding machine to contain this pressure

The injection molding process is capable of rapidly producing large quantities of parts with very high precision. Tolerances of a few thousandths of an inch are routinely achieved. With the right combination of material, part design and mold construction, even sub one thousandth inch tolerances can be achieved for small features.

The cost of injection molds can range from a few thousand dollars to hundreds of thousands of dollars.

5.1.1.2 **Materials:**

The materials used to construct injection molds range from aluminum to hardened steel:

Aluminum for simple low production prototypes.

The relative low strength of aluminum that makes it quicker to fabricate into molds likewise limits its useful life. Aluminum molds are typically intended to produce from a few thousand to a few hundred thousand parts with relatively simple features.
Prehardened tool steel for moderate production, more complex molds.
Prehardened tool steel molds are much stronger and more durable, yet still soft enough to be worked by conventional machining processes such as milling and turning. Prehardened tool steel molds are typically intended to produce from one hundred thousand to five hundred thousand parts, and can have a wide array of features such as slides and more intricate shapes that might break in an aluminum mold.
Hardened tool steel for high production, long life molds.
Hardened tool steel molds are the most durable and expensive because part way through fabrication their components are heat treated to achieve a hardness greater than can be machined. From that point on, the fabrication must continue using grinding and EDM processes.

Hardened steel molds are intended to produce one million or more parts. Their hardness enables them to resist wear from their own operation and the abrasion of the plastic material, particularly glass fiber reinforced materials. Hybrid construction is very common, where steel

parts are used in an aluminum mold to add strength to a slender feature, or parts of a steel mold are hardened to prevent wear at a rotating or sliding mold feature.

5.1.1.3 Molds:

Single cavity molds offer the lowest tooling costs and highest precision at the penalty of higher unit costs. Multi-cavity molds are utilized to increase capacity and lower unit costs.

Family molds, multi-cavity molds with different items together, offer both the lowest mold cost and low unit cost. However, they present other problems of matching the process conditions for each part and balancing supply when the product mix or yield at a later manufacturing step varies.

5.1.1.4 Engineering and Design:

The design of injection molds begins with a review of part specifications including: Aesthetics: color, clarity, high gloss, matte, special texture, etc. Material: strength, toughness, hardness, chemical and environmental resistance Interaction with mating parts: fits and tolerances Demand and unit cost goals

From this review process the mold design concept is evolved and decisions are made resulting in a mold specification:

Single, multiple cavity or family molds The grade of mold: aluminum, prehardened tool steel or hardened tool steel Material flow considerations Parting lines and gates Finish: high gloss, texturing, embedded text and graphics, etc. Accuracy and tolerances Cooling passages Ejection system Runners or runnerless system design

The next step is the actual design of the mold. Highly skilled designers using very complex and expensive computer software programs perform this. The design tasks include:

Modeling of the products and mold components in 3D. Mold flow analysis CNC tool path design and calculation Mold materials procurement list

Early in the design process, materials and components are ordered so that manufacturing can commence as soon as possible.

5.1.1.5 Manufacturing:

Once the design is completed manufacturing begins. Mold making involves many steps, most of which are very exacting work requiring highly skilled moldmakers. One mistake can ruin or cause major repair expense to a work piece that has undergone a series of manufacturing steps over several weeks. The processes employed in mold making include:

- o Milling and turning
- o Heat-treating
- o Grinding and honing
- o Electrical discharge machining
- o Polishing and texturing

To save cost, common mold components are purchased from suppliers. Frequently, outside services are required from subcontractors, which use specialty equipment such as thread grinding, etc.

When all of the parts are completed the next step is to fit, assemble and test the mold. All of the mold component parts must fit together precisely to achieve an aesthetic result on the product and for the mold to not wear out rapidly or break. The mold must be fluid tight to contain the molten plastic. Yet, at the same time the mold must have venting features added to allow the air to escape. The behavior of the plastic material when molded has been anticipated, however there can be some variance in the actual result. The mold must be tested to insure the products are correct and that the mold is performing properly. Where high accuracy is required, the mold may intentionally be made "metal safe" with the final adjustments coming after the first molding trial.

5.1.1.6 Conclusion:

As can be seen from the above, the engineering and creation of injection molds is a time consuming process. The work is demanding in terms of knowledge, skills and exacting attention to details. This will always be expensive, however this expense must be viewed in terms of what is achieved: Unsurpassed sophistication in part design and aesthetic appearance with low cost mass production.